ANALYSIS OF A SIMPLIFIED FRANGIBLE JOINT SYSTEM

Steven L. Renfro
The Ensign-Bickford Company
Simsbury, CT

James E. Fritz
The Ensign-Bickford Company
Simsbury, CT

Abstract

A frangible joint for clean spacecraft, fairing, and stage separation has been developed, qualified and flown successfully. This unique system uses a one piece aluminum extrusion driven by an expanding stainless steel tube. A simple parametric model of this system is desired to efficiently make design modifications required for possible future applications. Margin of joint severance, debris control of the system, and correlation of the model have been successfully demonstrated.

To enhance the understanding of the function of the joint, a dynamic model has been developed. This model uses a controlled burn rate equation to produce a gas pressure wave in order to drive a finite element structural model. The relationship of the core load of HNS-IIA MDF as well as structural characteristics of the joint are demonstrated analytically. The data produced by the unique modeling combination is compared to margin testing data acquired during the development and qualification of the joint for the Pegasus[*] vehicle.

Introduction

Frangible joints have been demonstrated as robust and contamination free separation systems for various spacecraft and launch vehicle stage and fairing separation. Typical frangible joint systems are initiated using mild detonating fuse (MDF) detonation products to expand an elastomeric bladder which then compresses dynamically against a formed stainless steel tube. The high pressure developed at the tube forces it to a more round shape in order to fracture an aluminum plate along a stress concentration groove. This fracture provides separation without fragmentation or contamination because the products are contained within the steel tube. A typical joint cross section is depicted in Figure 1.

Integrating this technology into new systems, with more challenging environmental conditions, could benefit from analytical modeling to properly configure each system. Understanding the mechanism required to sever the aluminum extrusion is crucial to meet new system requirements with full confidence.

The purpose of this report is to document The Ensign-Bickford Company's efforts to develop a simple analytical tool using widely available hydrodynamic and finite element computer codes.

Background

The ANSYS 5.0[*] finite element software allows for transient input to structures in the frequencies expected during a small damped detonation event. The frangible joint geometry is believed suitable for this type of analysis. To generate transient pulses for input into the finite element

model, simple one-dimensional hydrodynamic analysis is used.

For a one-dimensional Lagrangian model, a cylindrical geometry was assumed. The SIN[1] hydrodynamic code was used to solve conservation equations of momentum, mass, and energy. In order to use this information as input for the finite element model, individual or groups of cells were monitored to develop input equations for the finite element calculations.

Since the hydrodynamic analysis is one dimensional, it limits the amount of understanding developed regarding the specific stress state existing in the aluminum. Peak stress locations and probable points of secondary failure cannot be determined, and assumptions must be made for the stiffness and response characteristics. A two dimensional hydrodynamic analysis would assist in understanding these effects, but such analysis is time consuming, and requires access to sufficient hardware and software resources. Also, the hydrodynamic model is unable to account for a wide range of thermal loads and structural preloads in the parts, and cannot be used to evaluate stresses in the part due to events other than the explosive loading (i.e., flight loads, thermal response, assembly loading).

To understand the dynamic response of the frangible joint, an ANSYS 5.0 finite element model was created for use in a non-linear transient analysis. This analysis was used to determine the dynamic response of the aluminum when subject to transient loads driving the material above its yield strength. This methodology had been successfully used by The Ensign-Bickford Company to solve problems involving explosively and pryotechnically loaded structures. This paper represents the first time this technique used input developed by a one dimensional hydrodynamic analysis code.

Model Development

Using the SIN analysis, the critical areas were determined for input into the ANSYS 5.0 model. The timing and reaction of the shock waves incident and reflected from the interior steel wall result in two distinct types of relationships, both of which are decaying sinusoidal functions.

The area of the stress riser has extremely high initial amplitude which rapidly decays. This is consistant with the geometry present at this location. A thin layer of elastomeric material and a thin aluminum section bounding the steel do not support reflected pressure waves as well as the thicker off axis areas. Basically, the initial detonation front experiences a rapid ring down within the wall of the steel tube. The inside surface of the steel responds approximately as illustrated in Figure 2.

The second input function used is from a cross section at 45° from the stress riser. This relationship was similar in frequency to Figure 2 with a much lower initial amplitude. Figure 3 shows this relationship.

A logical choice for a third function is 90° to the separation plane. Most frangible joint designs use air gaps combined with thin silastic sections to control position of the MDF and to allow easy installation. If no air gaps are assumed, the resulting function resembles the data in Figure 3

with lower amplitude. Introducing the air gaps increases the difficulty of the hydrodynamic analysis without any real benefit. This third function was therefore not used for this simplified approach.

The source explosive used for this particular design is HNS-IIA. The equation of state for HNS is not currently available as part of the SIN database. Alternate explosive materials were used to bracket the response of HNS. The density, chemistry, detonation velocity, and Chapman-Jouget pressure were matched as closely as possible with candidate materials from the SIN database. Table 1 lists the explosives used and their properties compared to HNS.

To simplify the ANSYS model geometry, symmetric constraints are used along the notch edge and one half of the joint mounting flange. The length of the flange was shortened to reduce the number of degrees of freedom which needed to be incorporated into the model. This model is shown in Figure 4.

The transient loads are applied as pressure pulses along the interior of the aluminum. These loads have the same time profile as that predicted by the one dimensional hydrodynamic analysis, however input pressure amplitudes are reduced to achieve numerical stability. Unfortunately, this assumption is required, although it is expected that the results still allow development of an understanding of the aluminum response. The aluminum material (6061-T6) was assumed to act in an elastic-perfectly plastic manner. That is, once the yield strength of the material is exceeded, no additional load can be supported by that material. Plastic convergence is

achieved using the Modified Newton-Raphson method, based on a Von-Mises yield criterion. For the transient portion of the analysis, the Newmark time integration scheme is utilized, using Rayleigh damping with only mass matrix contributions (Beta damping). This applied damping is necessary in order to provide stability of the solution. However, a Beta term is chosen which ensures a low level of damping (0.05% or less) above 10,000 Hz.

For the initial time steps, the symmetry constraints are applied to both the top notch and the flange edges of the model. When sufficient stress levels are determined in the notch to induce section failure, this symmetry constraint on the notch is removed and the leg of the section is allowed to bend up and away from its initial position. This is done to simulate proper function of the joint during the explosive event.

Results of Finite Element Model

As part of the preliminary work performed using this model, simple static stress analysis (linear and non-linear) as well as modal analysis were performed to verify model integrity and to learn about the basic structural characteristics of the model. Some important data was gleaned from these runs, including the presence of a potential plastic hinge near the flange region of the aluminum structure. Additionally, the modal runs showed that the aluminum had its second, third, and fourth normal modes between 50 and 200 kHz. This was important information, since it showed that the aluminum is capable of dynamic elastic structural response near the input frequency of the shock pulse. The 2nd, 3rd, and 4th, mode shapes are shown in

Figure 5.

Once the simpler analysis had been run and verified with hand calculations, the more complex non-linear transient analysis was run. The input pulse was characterized as a shock pulse with a 1.5 μsec rise and a 1.5 μsec decay at the notch location. The magnitude of this pressure pulse was chosen to remain slightly below the yield point of the material (approximately 36 ksi) to avoid model stability problems. As noted above, this assumption needed to be made, however; much information about the dynamic response of the structure was still learned.

The final results are illustrated in Figure 6. During the rise time of the initial pressure pulse, the structure cannot significantly respond to the high frequency input. The structure simply transmits the shock wave through the material thickness. By the time the pulse is damped, the structure begins to significantly respond, and peak stresses in the notch exceed the allowable material strength. A plastic condition through the wall is reached. It is at this time that separation occurs, and the symmetric boundary condition along the notch edge is removed. After this time, the load is no longer applied and the inertial loads of the aluminum leg are all that is left driving the deflection. Obviously, the loading assumption is somewhat non-realistic; the shock wave applied to the aluminum will continue along the inner wall even after separation has occurred.

After this, the frangible joint is behaving as a cantilever beam with a fixed edge along the mounting flange. A plastic zone develops along much of the length of the flange wall. It is interesting to note that a plastic hinge develops in the bend region of the aluminum leg. This hinge location corresponds well to explosive over tests where a section of the aluminum became a flyer.

Finally, at 24 μsec, the leg has plastically deformed over its entire length. a plastic hinge occurs near the top of the extrusion, and model convergence is no longer possible using the elastic-perfectly plastic static strength allowable. The predicted deflection at this time is 0.103 inches. For the actual hardware, it would be expected that energy would be expended by bending at the plastic hinge until the impulse had been dissipated.

Discussion

The aluminum is capable of responding to the input shock pulse in the 2 to 3 μsec regime, suggested by the modal analysis and supported by the transient analysis. At approximately 3 μsec after the shock pulse has arrived at the interior of the aluminum stress riser, failure at the groove is expected to occur. There remains sufficient energy to severely deform the legs once the failure at the stress riser has occurred. A secondary plastic hinge forms at the bend joint near the mounting flange for this particular design.

The aluminum cross section is very efficiently dissipating the applied impulse once the stress riser failure occurs. In other words, plastic stresses do not localize and exist over much of the inner and outer surfaces of the aluminum.

All of these discussion items show good agreement with test specimen articles. No failures of this particular joint have

occurred which would disagree with the conclusions of this analysis.

The hydrodynamic analysis would provide much better resolution if a two dimensional model were used. Digital resolution of individual cell results could be used as forcing function for the finite element techniques.

Although not specifically addressed by this paper, the one dimensional hydrodynamic analysis combined with the two dimensional ANSYS analysis shows good promise for evaluation of the effects of thermally induced strains and launch load induced stresses. A two dimensional hydrodynamic input would further enhance the ability of this technique to simulate flight functional conditions.

References:

1)   Charles L. Mader; Numerical Modeling of Detonations; University of California Press; 1979; pp. 310 - 332.

2)   B.M. Dobratz; LLNL Explosives Handbook; UCRL-52997,
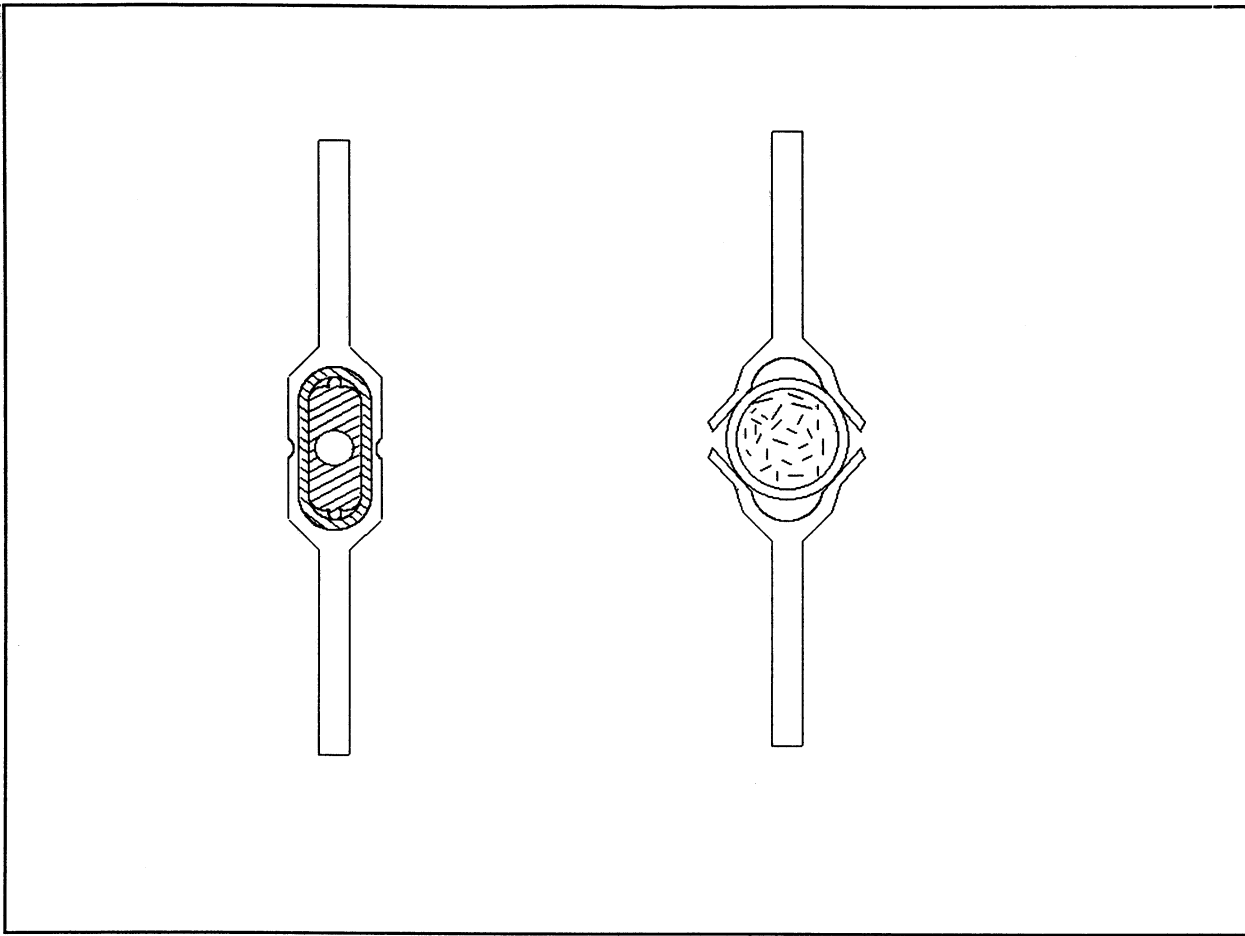
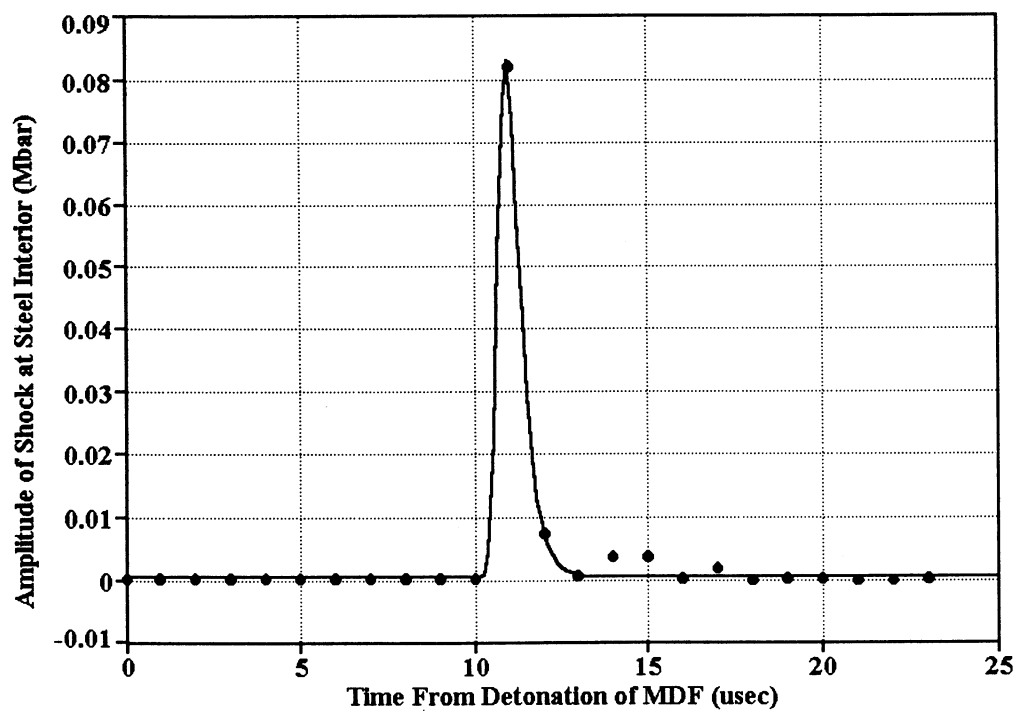**Figure 1** Frangible Joint Before and After Function

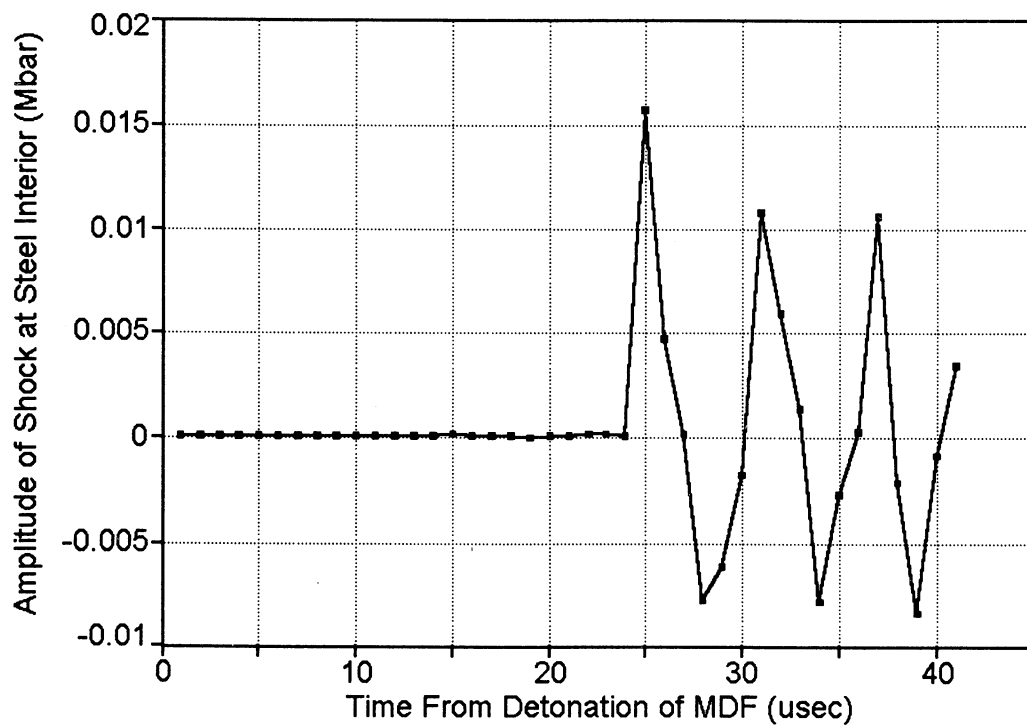**Figure 2** Pressure Time History at Interior of Steel Tube at Stress Riser

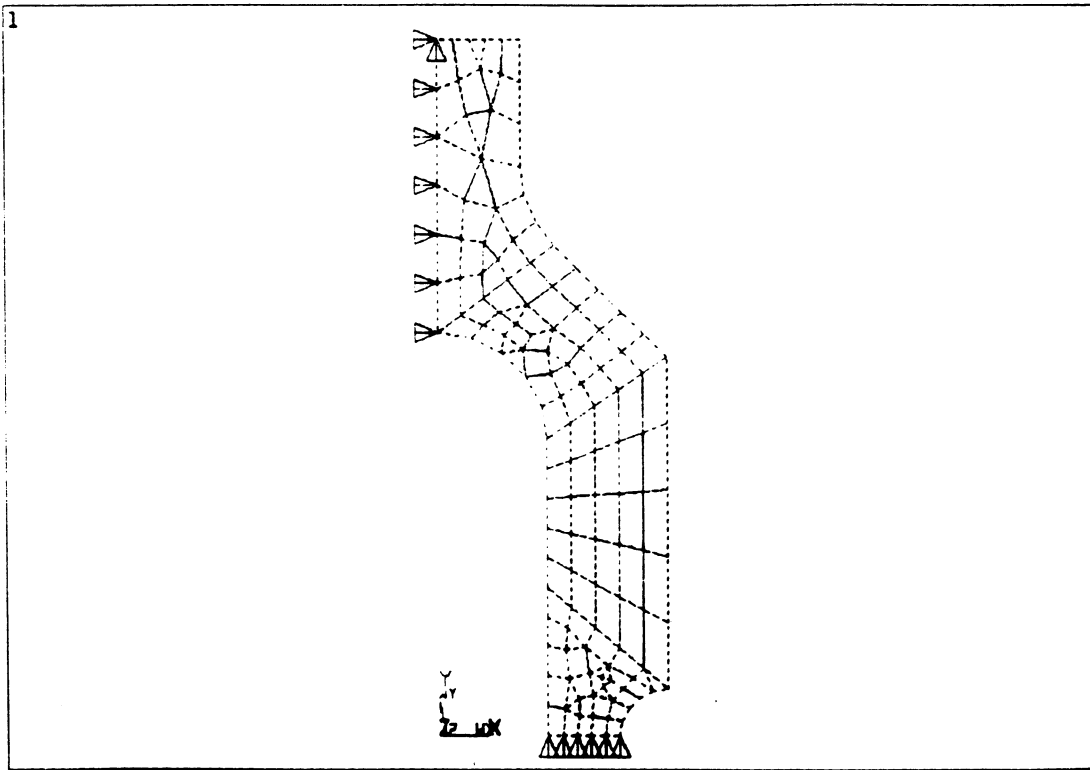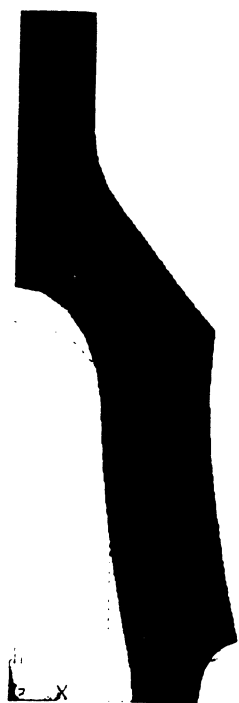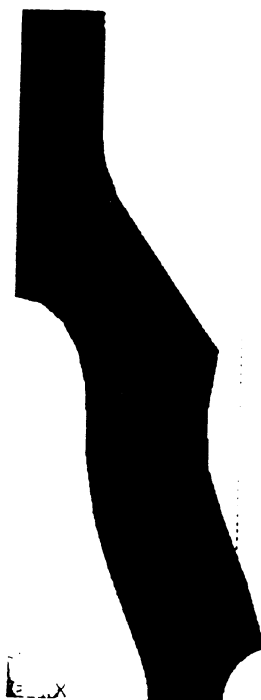**Figure 3** Pressure Time History at 45° From Stress Riser
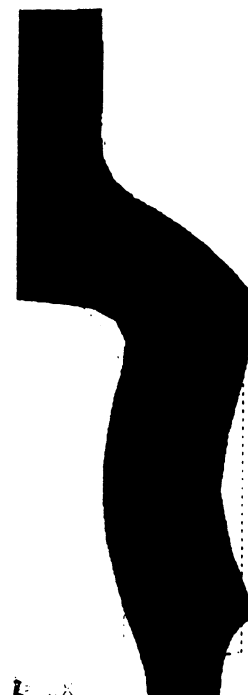
Figure 4 ANSYS Finite Element Model

MODE 2
52.827 HZ

MODE3
102.348 HZ

MODE 4
154.759 HZ

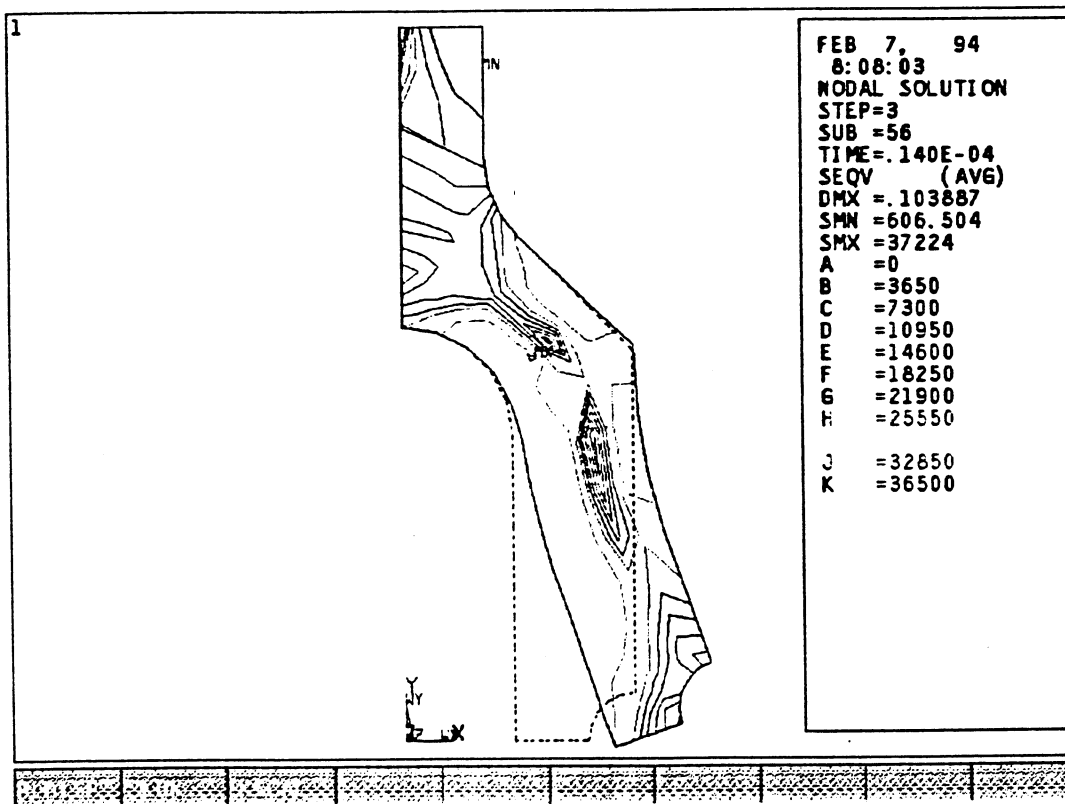**Figure 5** ANSYS Finite Element Model Elastic Normal Modes

Figure 6 Results of Hydrodynamic and FEA Combined Model

Table 1. Explosive Properties Used to Bracket HNS Performance[2]

Material	Chemical Formula	Density (g/cm^3)	Detonation Pressure (kbar)	Detonation Velocity (mm/μsec)
HNS	$C_{14}H_6N_6O_{12}$	1.60	200	6.80
TATB	$C_6H_6N_6O_6$	1.88	291	7.76
TNT	$C_7H_5N_3O_6$	1.63	210	6.93